Analysis of Models Under Location Uncertainty within the Framework of Coarse Large Eddy Simulation (cLES)

Pranav Chandramouli, Dominique Heitz, Etienne Mémin, Sylvain Laizet

Analysis of Models Under Location Uncertainty within the Framework of Coarse Large Eddy Simulation (cLES)

Pranav Chandramouli[1], Dominique Heitz[1,2], Etienne Mémin[1], and Sylvain Laizet[3]
[1] INRIA, Fluminance group, Campus universitaire de Beaulieu, F-35042 Rennes Cedex, France
pranav.chandramouli@inria.fr
[2] Irstea, UR OPAALE, F-35044 Rennes Cedex, France
[3] Department of Aeronautics, Imperial College London, London, United Kingdom

Large Eddy Simulations (LES) have become common place in the current research scenario with increasing computational resources. However, constraints still limit the application of LES in a variety of scenarios: high Reynolds (Re) number flows, complex geometry flows, or flows involving complicated wall boundary layers. While the last scenario is limited due to physical aspects of the model, the first two can be rectified by reducing the computational cost of performing LES. An immediate foreseeable solution is to reduce the number of computational points in the simulation. However, this leads to a stark decrease in accuracy for an LES model. Complex methodologies have been developed to negate this decrease such as hybrid RANS-LES models: using RANS model (respec. LES model) for coarse grid (respec. fine grid) regions. In this study, the focus is on physical behaviour characterisation of novel models under location uncertainty [1] in a coarse mesh construct. Analysis and comparisons with the performances of classic LES models, decreasing in accuracy with increasingly coarse meshes, are conducted. The models under location uncertainty originate from the stochastic mass and momentum conservation equations which are derived using stochastic calculus. Similar to a filtered NS equation for LES, the stochastic version contains a sub-grid scale dissipation terms – this term is fully specified; there is thus no need to rely on the additional Boussinesq viscosity assumption. It also contains a sub-grid scale velocity bias term acting on the advection component - this term is related to a phenomenon termed *'turbophoresis'* in literature and is usually not taken into account in classical sub-grid modelling. Both terms are characterised by the small-scale velocity auto-correlation ($\mathbf{a} = \sigma\sigma^T$) which requires modelling. While a Smagorinsky-like model under location uncertainty (StSm) can be envisaged (through a local isotropy assumption), the performance of these models excels when a local variance based **a** (StSp – spatial variance; StTe – temporal variance) is realised. The performance of these models is compared with the classic (Smag) and dynamic Smagorinsky (DSmag) models, and the Wall-Adaptive Local Eddy viscosity (WALE) model. Two well-studied flows, namely wake flow around cylinder at Re = 3900, and channel flow at $Re_\tau = 395$, are used to analyse the performance of the models under a coarse resolution with reference statistics from [2] for wake flow (see fig 1.) and [3] for channel flow (see fig 2.). The statistical correlations are shown to be better even at low resolutions for the models under location uncertainty while the classical LES models are either inaccurate or numerically unstable. A flow with well-resolved vortices is observed with the models under location uncertainty and they also capture the important turbulent characteristics of a given flow better than the classical models.

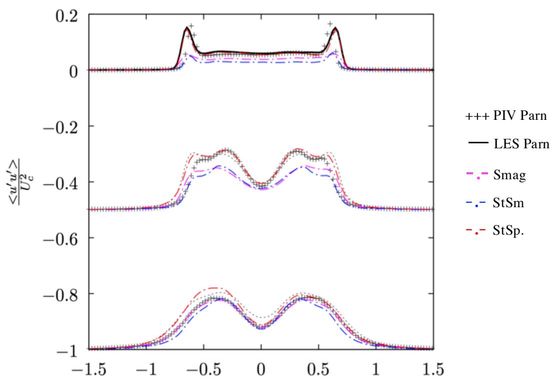

Fig. 1: Streamwise rms velocity at 1.06D, 1.54D, and 2.02D in the wake of the circular cylinder.

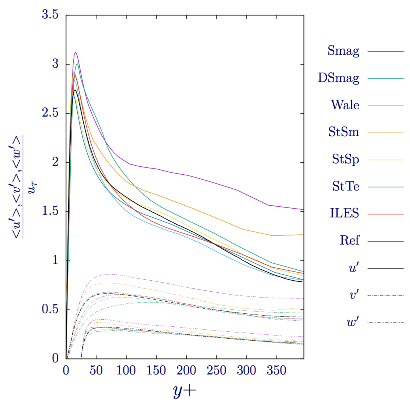

Fig. 2: Velocity fluctuation statistics for channel flow at $Re_\tau = 395$

References

[1] Mémin E. Fluid flow dynamics under location uncertainty. Geophysical & Astrophysical Fluid Dynamics, 108, 119-146, 2014.
[2] Parnaudeau P, Carlier J, Heitz D and Lamballais E. Experimental and numerical studies of the flow over a circular cylinder at Reynolds number 3900. Physics of Fluids (1994-present) 20, no. 8: 08510, 2008.
[3] Moser R. D., Kim J., and Mansour N. N. Direct numerical simulation of turbulent channel flow up to Re=590. Physics of Fluids, 11(4):943-5, 04, 1999.